Flip for **Tutorial**

User Guide

Mastercam® PERFORMANCE PACK

by

CIMCO Integration

When Reliability Matters

License Information

Information in this document is subject to change without notice and does not represent a commitment on the part of CIMCO Integration. The software described in this document may be used or copied only in accordance with the terms of the license. The purchaser may make one copy of the software for a backup, but no part of this user manual may be reproduced, stored in a retrieval system, or transmitted in any form or by any means electronic or mechanical, including photocopying and recording for any purpose other than the purchaser's personal use, without prior written permission from CIMCO Integration.

TERMS OF USE FOR SOFTWARE

Software: Mastercam HSM Performance Pack
Date: July 2003

Notice:
CIMCO Integration reserves the right to make improvements to the Mastercam HSM Performance Pack Software at any time and without notice.

Software License

You have the right to use the number of licenses of the enclosed program, which you have bought from CIMCO Integration. You may not distribute copies of the program or related documentation to any persons or companies. You may not modify the program or related documentation without the prior written consent of CIMCO Integration.

Disclaimer of all Warranties and Liability

CIMCO Integration makes no warranties, either express or implied, with respect to the software, its quality, performance, merchantability, or fitness for any particular purpose. The entire risk as to its quality and performance is with the buyer. Should the Mastercam HSM Performance Pack software prove defective following its purchase, the buyer (and not

CIMCO Integration, its distributor, or its retailer) assumes the entire cost of all necessary servicing, repair, of correction and any incidental or consequential damages. In no event will CIMCO Integration be liable for direct, indirect, or consequential damages resulting from any defect in the software, even if CIMCO Integration has been advised of the possibility of such damages. Some jurisdictions do not allow the exclusion or limitation of implied warranties or liability for incidental or consequential damages, so the above limitation or exclusion may not apply to you.

NOTICE: The accompanying software is confidential and proprietary to CIMCO Integration. No use or disclosure is permitted other than as expressly set forth by written license with CIMCO Integration.

Contacting CIMCO Integration

Address:
CIMCO Integration I/S
Maglebjergvej 6
DK-2800 Lyngby
Denmark

Phone: +45 4585 6050
Fax: +45 4585 6053
E-mail: info@cimco-software.com
Web: http://www.cimco-software.com

Mastercam HSM Performance Pack User Guide

Installing Mastercam HSM Performance Pack

You must be logged in as an administrator to install Mastercam HSM Performance Pack.

Install the .msi file. If you choose the default settings, the module will be installed under "Mastercam HSM Performance Pack" in the "Program Files" folder.

Start Mastercam (Mill 9.1) and go to Main menu → Screen → Configure → NC Settings → Tool Display. Here you must deactivate "Repaint toolpath", you then exit by clicking “OK” twice, and accept to save your changes. It is best to repeat this for all your configuration files.

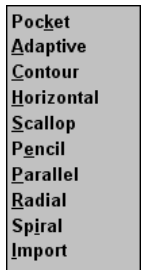
Start the C-hook (named "hsm.dll") from the installation folder (default "C:\Program Files\Mastercam HSM Performance Pack") by pressing Alt-C in Mastercam, and navigating to that folder. The C-hook will copy the necessary files to the Mastercam C-hook folder.

Start the C-hook (named "hsm.dll") from the Mastercam C-hook folder as above. Note, that it is important that this step is done using “Alt-C” and navigating to the folder, otherwise the search path for C-Hook files will not be correctly reset in Mastercam.

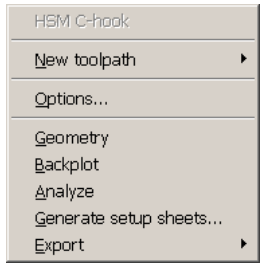
If you are using the MCUtils C-Hook, then icons for the Mastercam HSM Performance Pack toolpaths can be added to the toolbar on the right of the operations manager by right clicking in the toolbar and selecting “Options”.

Using Mastercam HSM Performance Pack

Mastercam HSM Performance Pack is started from Mastercam with Alt-C, where you then select HSM.dll. You are then presented with this menu of machining strategies:



When Mastercam HSM Performance Pack has been run once during the current Mastercam session, you can also access a menu from the operations manager in Mastercam. If you right-click in the operations manager while holding the Ctrl and Shift buttons down, you will get this menu:



New toolpath opens a submenu with the machining strategies, allowing you to create a new toolpath of the desired type.

Options opens the options dialog. See “The options dialog” on page 52.

Geometry opens CIMCO Inspect, or imports the part geometry in CIMCO Inspect if CIMCO Inspect is already open.

Backplot opens CIMCO Inspect, or imports the selected toolpath(s) in CIMCO Inspect if CIMCO Inspect is already open.

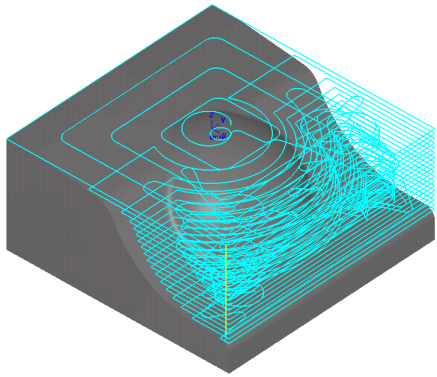
Analyze brings up an analysis of the selected toolpath.

Generate setup sheets creates setup sheets for the selected operations.

Export allows you to select an operation, which you can then generate a setup sheet for, or you can export the parameters or toolpath as an XML file.

Machining strategies

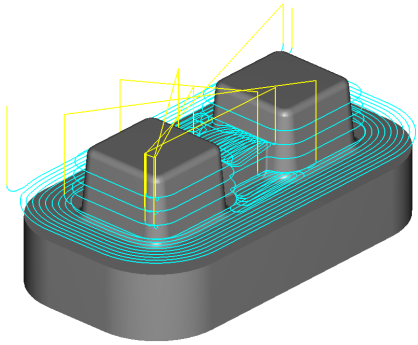
Pocket



This is the first of the main roughing operation. It can be thought of as an extension of the contour machining operation because it makes similar paths with constant Z height, but creates the offset clearing paths within them as well. All paths are smoothed with no unnecessary corners, and the linking moves make helixes and ramps between levels. The areas are always cleared as though machining in a pocket.

There is the also a rest roughing feature where you can select a previous tool or operation, which you have roughed with first, or a STL stock model. Then the contour clearing is only done in the areas where the previous tool did not fit.

Adaptive clearing

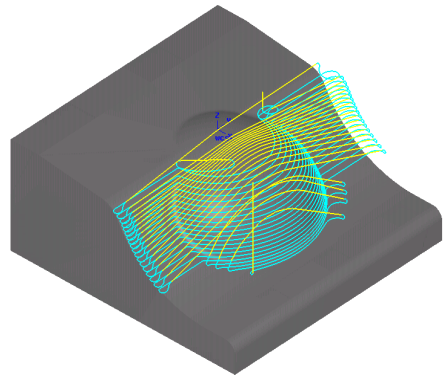


Adaptive Clearing avoids full-width cuts when machining by generating a toolpath that progressively shaves material from the stock model in trochoidal motions. The unique feature is that the cutting conditions are constant but the radius is always as large as possible, so the result is a more constant feed rate and less tool vibration. With a traditional roughing algorithm there is always a mixture

of side and slot milling throughout the operation. But, since Adaptive Clearing guarantees that there will never be any full-width slot milling, you can use the recommended side milling cutting conditions from your tool manufacturer's catalogue. Since the toolpaths from adaptive are always flat areas with constant load, the feed and speed can be increased to 150-200% of the recommended values. Further, as the toolpaths are using side cutting conditions, the stepdown for major steps is usually at least equal to the tool diameter, so the wear on the tool is along most of the flute instead of only the corner of the tool, prolonging tool life.

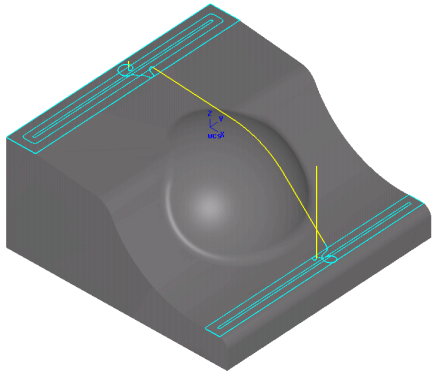
The tool plunges outside the stock and machines the part from outside in whenever possible, only making ramp/helix entry moves when entering cavities or when the tool boundary prevents plunging outside the part.

Contour machining



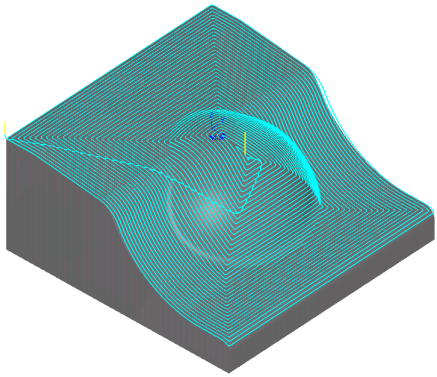
This operation creates machining passes with constant Z height, which run along contours of the part.
The passes are linked from higher to lower, and the stock material remaining on the part is assumed to be less than one cutter radius - a common requirement for finishing operations to work.
The cutting paths are smoothed in the corners, but by a horizontal allowance (deviation) rather than a radius. This means that acute corners tend to have very small smoothing radii in them, because the arc would otherwise pull back from the corner too far. This represents a compromise between the requirements of high speed tooling, and the need to actually follow the surfaces of the part.

Horizontal clearing



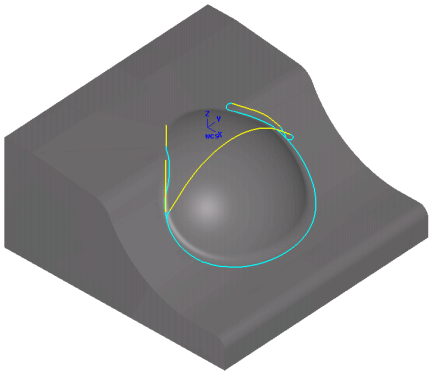
This operation detects the perfectly flat areas of the part and makes an effective contour clearing operation over them. When the flat area is shelved, the cutter goes out beyond the edge for a better result, in the areas where the part is lower beyond the edge.

Scallop finishing



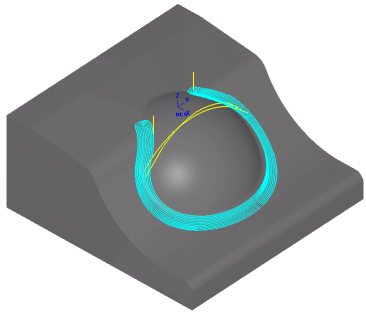
Sometimes called constant stepover passes. The first pass follows the boundary in the plane projected onto the surface. Each subsequent pass is offset inwards along the surface to remain at a constant distance. The passes will follow up- and down-sloped and vertical walls to maintain this distance.
This operation will also give you rest area machining if you enable the previous tool parameters.

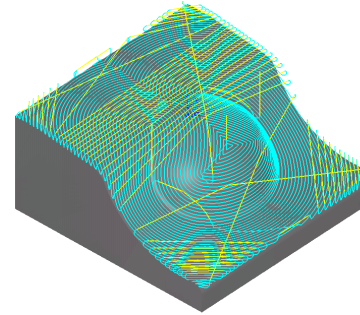
Pencil machining



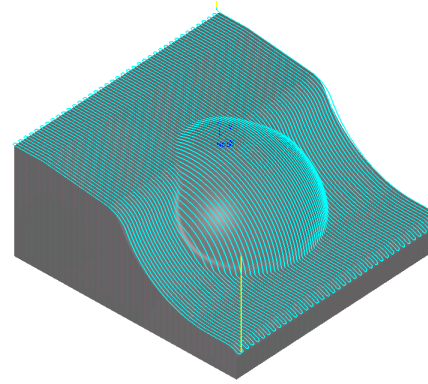
The defining parameter for this operation is the bitangency angle. The system will run the cutter along all the paths where it would be making two simultaneous contacts with the part with more than this angle in difference.
The overthickness enables you to detect a set of pencil passes for a

cutter larger than the one you specify, and project the passes into the surface. This makes it possible to find paths along a corner fillet where the radius is slightly greater than the cutter radius.
If you specify a number of offsets greater than zero, you will get passes on either side of each pencil pass (below to the left), or you can set unlimited offsets for machining the entire surface (below to the right).



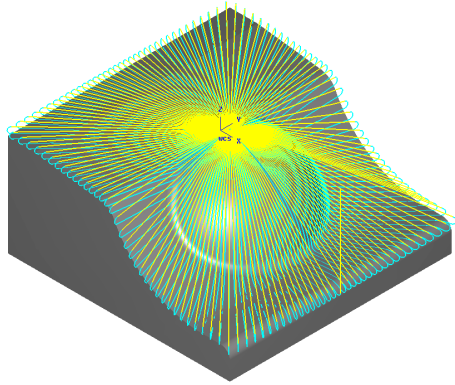


Parallel finishing



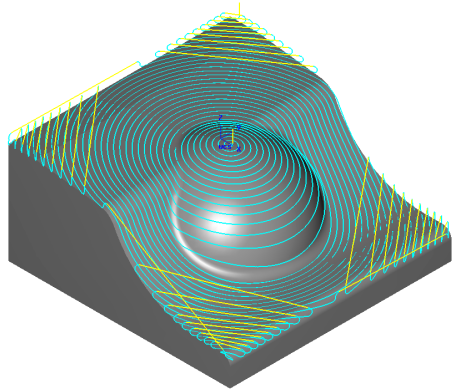
This is the simplest of the machining strategies. The passes created are parallel in the XY plane, following the surfaces in the Z axis. It is possible to specify the angle, such as 0º for parallel to the X axis or 90º for parallel to the Y axis (or any other angle), as well as the stepover in the horizontal direction. The passes can be made in one direction, retracting and returning at rapid, or they can be cut in a zig-zag pattern.

Radial finishing



This creates passes along the radii of an arc, similar to the spokes of a wheel, which are projected down on the surfaces. The passes are limited by an inner and outer radius (the inner radius can be zero), and can also be limited by a starting and ending angle. The passes can be zig-zag between the outer and inner radius, going from inside out or from outside in, or they can be kept either down milling or up milling, as desired.

Spiral finishing



This creates a spiral path, which is projected down on the surfaces. This is most useful in round, relatively flat areas.

Import

This allows you to import an operation from a saved XML file. The saved operation can be any type of operation generated by Mastercam HSM Performance Pack. After importing an operation, it is necessary to select the geometry for the operation, and the operation can be edited if necessary.

Defining the toolpaths

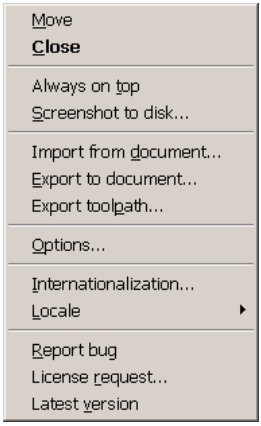
After selecting a machining strategy, you are prompted to select surfaces and solids, and then to select a boundary. If no boundary is selected, then the limits of the part are set as the boundary. Once the geometry has been selected, the toolpath dialog is seen, where the two most important tabs are the 'Tool' tab, and the tab representing the machining strategy selected at the start.

Right clicking in a dialog has different results, depending on whether the cursor is in a field or not.

In a field, a menu appears with relevant options for the field, such as selecting the coordinate from an element in the geometry window, or browsing for a file. If the entry for the field is a file, there is also a "recent files" list to choose from.

Outside a field, the options are to set the user defaults for this page or all pages to the current values, and to reset this page or all pages to either the user defaults or the built in defaults

Clicking on the CIMCO Integration logo in the upper left corner of the dialog window brings up this menu:



Move allows you to move the dialog to another position on the screen.

Close closes the dialog without saving any changes.

Always on top toggles, whether the dialog is kept on top of all other open windows.

Screenshot to disk lets you save a BMP image of the current dialog.

Import from document imports the values from a XML document, saved using *Export to document* below.

Export to document can export the values from the operation to a XML document, and can create a setup sheet for the operation. The XML file is useful for accurately recording settings for later reuse or reporting bugs, for example. If a setup sheet is desired, the user must select "HTML Document Setup Sheet (*.html)" from the "Save as type" dropdown menu.

Export toolpath can export the toolpath to a .cld cutter location file, a nci file or a XML document.

Options is used to access the options dialog, described at the end of this manual, where global options for Mastercam HSM Performance Pack are set.

Internationalization opens the translation utility, where you can make your own translation of the dialogs and messages in Mastercam HSM Performance Pack.

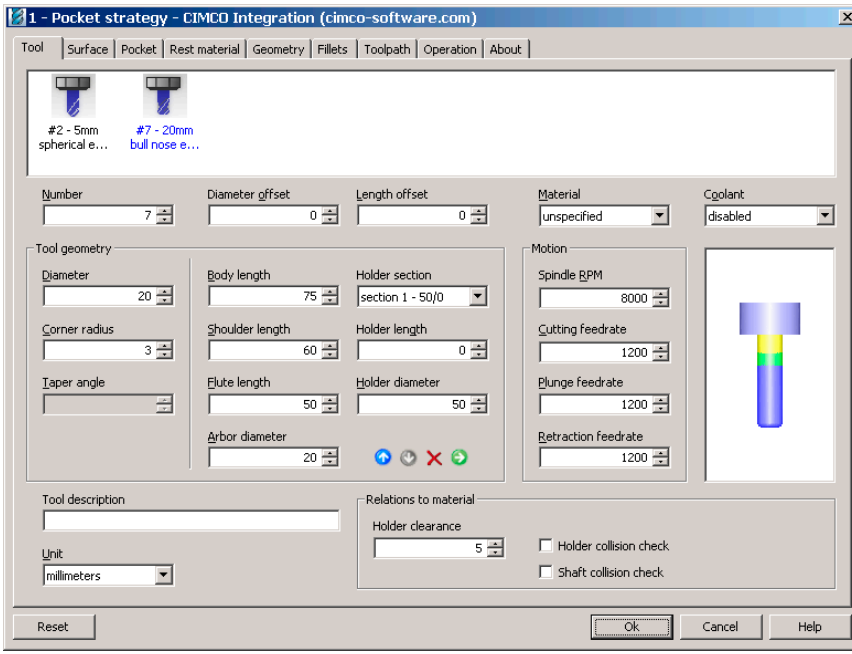
Locale opens a sub menu, where you can select the language you wish to see Mastercam HSM Performance Pack in. This sub menu contains both the predefined languages that were installed with Mastercam HSM Performance Pack, and any new translations you have made with the *Internationalization* option above.

Report bug opens your e-mail client with a bug report e-mail almost ready to be sent. You will only need to fill in a description of the bug, and attach any files necessary for documenting it.

License request: This opens a license request dialog, which you can use if you need to send a request to your dealer, for changes in your contact information or for additional licenses.

Latest version reports the version number and release date of the latest version of Mastercam HSM Performance Pack available for download. You can then view the change log for the latest version and download the latest version if desired. This only works if the PC is connected to the internet, and the newer version downloaded is not automatically installed, it must be manually installed after download to be used.

The tool tab



In the large area, the tool is selected, or a new tool can be created from the right click menu. The fields below are automatically updated from the tool selected above. The tool drawing to the right is updated each time a field is exited, to reflect the changed value. The right click menu is also used to access the HSM tool library, described below.

The flute length doesn't have an effect on the shape of the cutter paths unless a taper angle is applied. Tapered tools add the shape of a cone of the given taper angle from the vertical axis. The bottom of the (truncated) cone is tangential to the shape at the tip, and the top is at the flute length above the tip. For tapered tools, the tool definition is straightforward for ball nosed and flat bottomed tips, but for bull nosed tips the standard convention is more complicated: the shaft diameter refers to the diameter of the bottom of the conical surface and not the diameter of the virtual toroidal shape that would be within the tapered shape.

In the *Holder section* area, the shape of the holder can be defined and edited. The *Holder section* dropdown menu selects the section to be edited, which is highlighted to the right, and the size of the section can be edited in the *Holder length* and *Holder diameter* fields. The tool and holder image to the right is dynamically updated with the changes. There are also these four buttons:

 Move up to the next higher holder section.

 Move down to the next lower holder section.

 Remove the selected holder section.

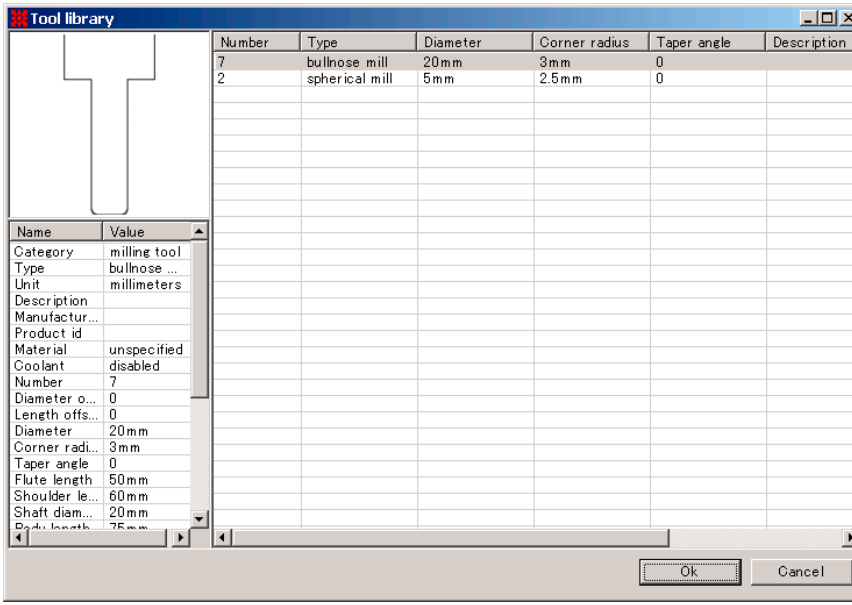
 Add a holder section to the top of the holder.

In the *Relations to material* area, it is possible to set holder and shaft collision checks, and a clearance distance for the holder check. These checks require additional calculation time when the toolpath is generated, but ensures that the shaft and holder do not collide with the part if used.

Right clicking outside a field also allows the user to set the capabilities of the tool (roughing/finishing) and the direction of spindle rotation (clockwise/counter-clockwise).

HSM tool library

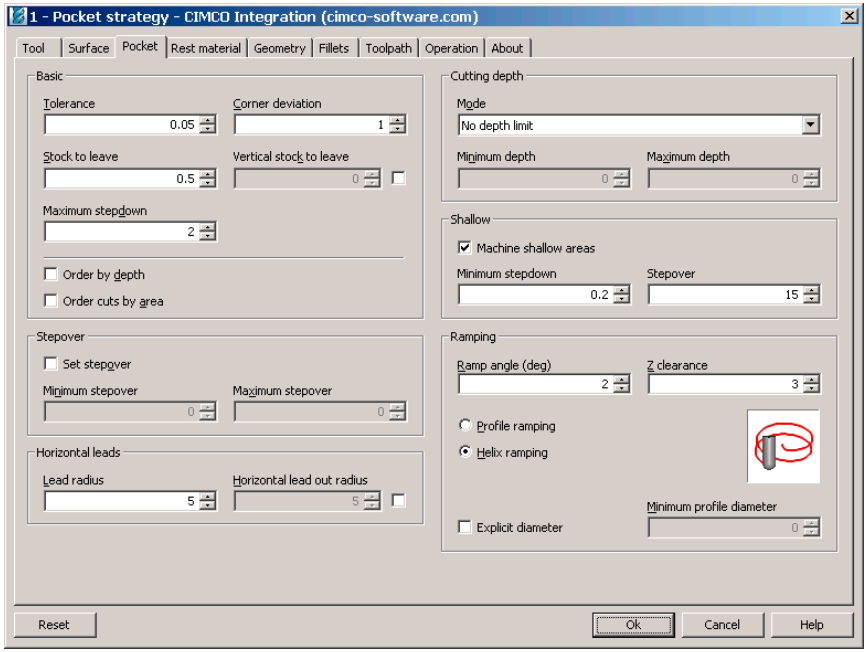
The HSM tool database is accessed through the right click menu on the tool tab. The tool library dialog looks like this:



To the left, the selected tool can be viewed. A tool can be edited by double-clicking it in the list, or by right-clicking it and selecting edit. Right clicking in the list also gives other options, such as adding a new tool, importing the tools in the current part file, duplicating tools, removing tools, changing database, etc.

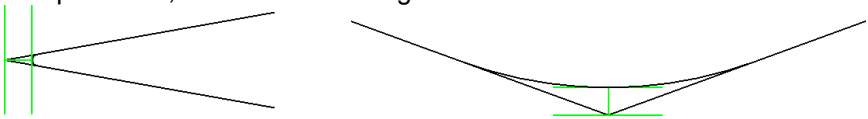
The machining strategy tab

Pocket



Tolerance is the maximum deviation from the surface, which the contact point of the cutter will have while machining. This tolerance is before arc filtering is applied, if used. See also *Global filtering* on the operation tab.

Corner deviation describes the smoothing done in corners. To create a smooth toolpath, arcs are inserted in the corners of the toolpath, keeping the smoothened toolpath within the deviation allowance defined by the corner deviation. As can be seen below, this creates small radius arcs in sharp corners, while it creates large radius arcs in shallow corners.



Stock to leave is the amount of stock to leave on the surfaces. By checking the checkbox to the right of *Vertical stock to leave*, it is possible to set different amounts of stock to be left in the horizontal and vertical directions.

Maximum stepdown is the step in the Z direction.

Order by depth: Checking this checkbox will cause the toolpath to be linked in order of Z depth, so an entire Z depth is machined before the next z depth, even if there are several areas. This function should be used if there are thin walls between areas that have to be machined, or if it is in a deep and narrow pocket.

Order cuts by area: Checking this checkbox will cause the toolpath to be linked so passes in one area will be finished before the tool moves on to the next area. This will minimize the number of retracts, but may cause problems if there are thin walls between areas that have to be machined, or if it is in a deep and narrow pocket.

Stepover: It is possible to specify a maximum step in XY per pass, instead of allowing Mastercam HSM Performance Pack to calculate the maximum step possible while still removing all material. It is usually best to allow Mastercam HSM Performance Pack to calculate the maximum step possible while still removing all material, as it is often impossible to avoid machining with the full diameter of the tool in some areas anyway.

Horizontal leads: The *Lead radius* here is the radius of the lead in/out from the toolpath, in the horizontal plane. By selecting the checkbox adjacent to *Horizontal lead out radius*, it is possible to set a different radius for the lead out from the toolpath in the horizontal plane.

Cutting depth: Here it is possible to specify depth limits for the toolpath, useful for machining deep pockets, where the upper areas are to be machined with a shorter tool, and the lower areas with a longer tool. The depth limits are to the tip or center of the tool (as selected here), even if *Boundaries confine the tool contact points* is selected on the geometry tab.

Shallow: Selecting *Machine shallow areas* enables the check for shallow and horizontal areas, adding Z levels to ensure that there is not left

excessive material above them. Selecting *Machine shallow areas* will also add a Z level for each horizontal area above the bottom of the pocket, machining that area to leave only *Stock to leave* above it.

Minimum stepdown: This is the minimum step down between two Z levels. This setting overrides the *Stepover* setting for shallow machining, described next, so no two Z levels will be closer to each other than the *Minimum stepdown*.

Stepover: This is the maximum stepover between two Z levels, i.e. the maximum distance in XY from the edge of one Z level to the edge of the next Z level. The *Minimum stepdown* setting described above overrides this.

Ramping: This describes the entry moves for each Z level.

Ramp angle (deg.): This is the angle from horizontal, which the center of the tool moves at during the entry ramping.

Z clearance: The ramping entry starts this distance above the top of the stock before the first Z level of each pocket.

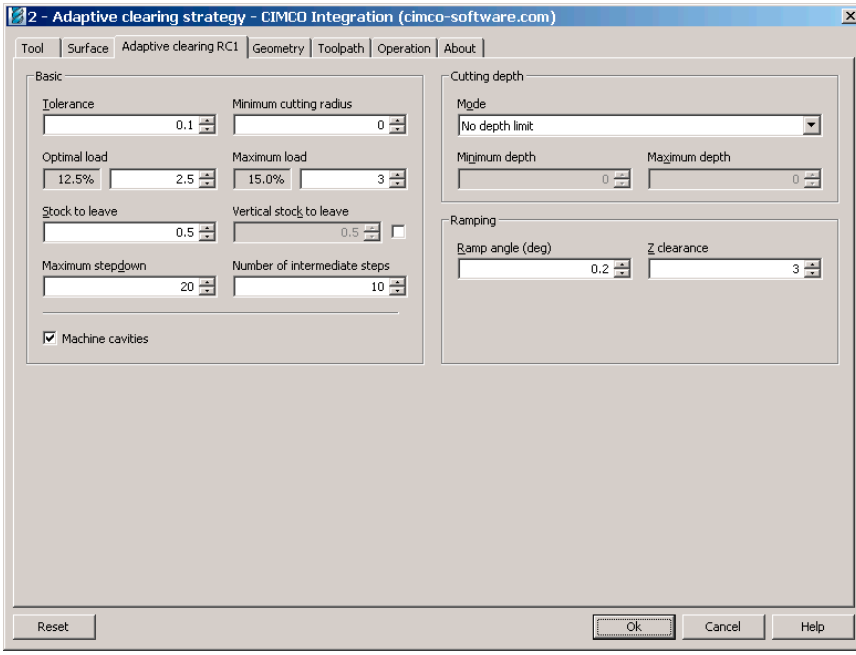
Profile ramping: The tool will follow the first pass of the next Z level as it ramps down.

Helix ramping: The tool will ramp down in a helix (spiral), where the radius of the helix is calculated automatically, based on the tool. If there is not room for a helix, then the tool will follow the first pass of the next Z level as it ramps down.

Explicit diameter: This option allows a minimum size for the entry profile to be set.

Minimum profile diameter: This is the minimum profile size for the entry ramping. It is measured along the longest direction of the profile. If *Helix ramping* is selected above, and there is not room for the helix, then *Profile ramping* is used. If *Profile ramping* is selected (or helix ramping was discarded), then the first pass of the new Z level that is larger than the *Minimum profile diameter* will be used for ramping.

Adaptive



Tolerance is the maximum deviation from the surface, which the contact point of the cutter will have while machining. This tolerance is before arc filtering is applied, if used. See also *Global filtering* on the operation tab.

Minimum cutting radius defines the minimum radius of the arcs used for the trochoidal motion in sharp corners and narrow channels. This should be set to 0 or a value that is at least twice the tolerance.

Optimal load and *Maximum load* defines the width of the cut used. The *Maximum load* should be set to the maximum load for the cutter, while the *Optimal load* should be set a little under that (no less than ¾ of maximum load). The optimal and maximum load is also shown in percent of the cutter diameter for the user's convenience.

Stock to leave is the amount of stock to leave on the surfaces. By checking the checkbox to the right of *Vertical stock to leave*, it is possible to set different amounts of stock to be left in the horizontal and vertical directions.

Maximum stepdown is the major stepdown, usually equal to or greater than the tool diameter. The adaptive toolpath makes a full Z level cut at a major stepdown before proceeding to make the intermediate steps between that major stepdown and the prior (higher) major stepdown.

Number of intermediate steps is the number of intermediate steps made between each major stepdown. When the adaptive toolpath has made a major stepdown, it proceeds to make the intermediate steps between that major stepdown and the prior (higher) major stepdown from lowest Z and up. The intermediate steps are made as contour passes if the material to be removed is within the *Maximum load*, otherwise the intermediate steps makes trochoidal motion like a major step.

Machine cavities is used to select, that cavities (areas that cannot be reached from the outside of the part without gouging or exceeding the *Maximum load*) should also be machined. When selected, the tool enters the cavities using the settings under *Ramping*.

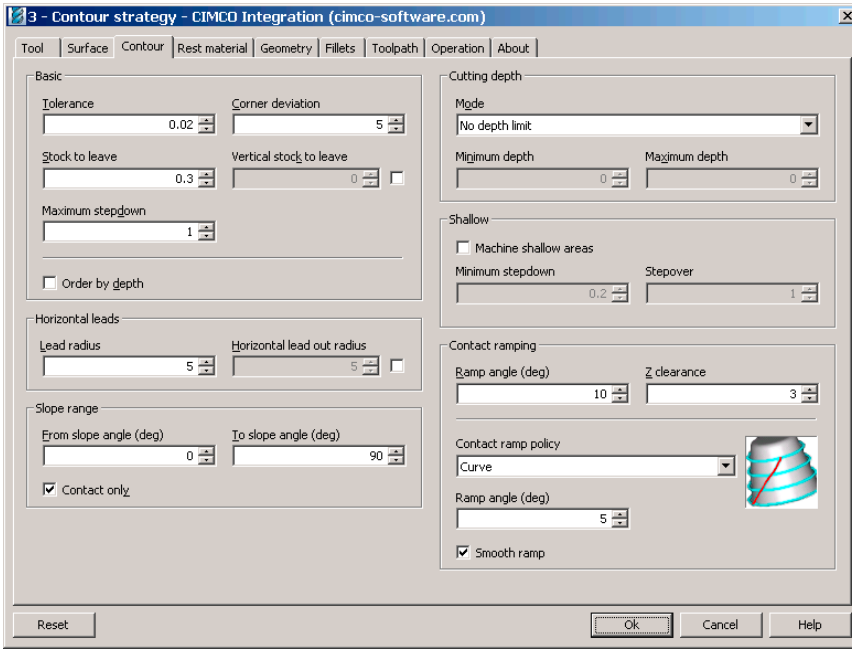
Cutting depth: Here it is possible to specify depth limits for the toolpath, useful for machining deep pockets, where the upper areas are to be machined with a shorter tool, and the lower areas with a longer tool. The depth limits are to the tip or center of the tool (as selected here), even if *Boundaries confine the tool contact points* is selected on the geometry tab.

Ramping: This describes the entry moves for each major Z level inside a cavity.

Ramp angle (deg.): This is the angle from horizontal, which the center of the tool moves at during the entry ramping. When using the cutting data normal for an adaptive toolpath, the ramp angle must be very low, a value of 0.2 degrees is not unusual.

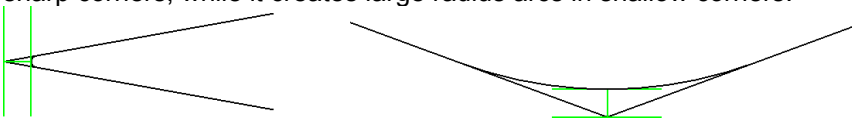
Z clearance: The ramping entry starts this distance above the top of the stock before the first Z level of each pocket.

Contour



Tolerance is the maximum deviation from the surface, which the contact point of the cutter will have while machining. This tolerance is before arc filtering is applied, if used. See also *Global filtering* on the operation tab.

Corner deviation describes the smoothing done in corners. To create a smooth toolpath, arcs are inserted in the corners of the toolpath, keeping the smoothened toolpath within the deviation allowance defined by the corner deviation. As can be seen below, this creates small radius arcs in sharp corners, while it creates large radius arcs in shallow corners.



Stock to leave is the amount of stock to leave on the surfaces. By checking the checkbox to the right of *Vertical stock to leave*, it is possible

to set different amounts of stock to be left in the horizontal and vertical directions.

Maximum stepdown is the step in the Z direction.

Order by depth: Checking this checkbox will cause the toolpath to be linked in order of Z depth, so an entire Z depth is machined before the next z depth, even if there are several areas. This function should be used if there are thin walls between areas that have to be machined, or if it is in a deep and narrow pocket.

Horizontal leads: The *Lead radius* here is the radius of the lead in/out from the toolpath, in the horizontal plane. By selecting the checkbox adjacent to *Horizontal lead out radius*, it is possible to set a different radius for the lead out from the toolpath in the horizontal plane.

Slope range: Only areas where the slope angle of the surfaces is within these limits will be machined.

Contact only: If this checkbox is checked, toolpath segments where the tool is not in contact with the surfaces will be filtered out. With this checkbox not checked, there will also be contour passes along the tool boundary.

Cutting depth: Here it is possible to specify depth limits for the toolpath, useful for machining deep pockets, where the upper areas are to be machined with a shorter tool, and the lower areas with a longer tool. The depth limits are to the tip or center of the tool (as selected here), even if *Boundaries confine the tool contact points* is selected on the geometry tab.

Shallow: Selecting *Machine shallow areas* enables the check for shallow areas, adding contour passes for a better surface finish.

Minimum stepdown: This is the minimum stepdown between two contour passes. This setting overrides the *Stepover* setting for shallow machining, described next, so no two contour passes will be closer to each other in the Z direction than the *Minimum stepdown*.

Stepover: This is the maximum stepover in XY between two contour passes. The *Minimum stepdown* setting described above overrides this.

Contact ramping sets the motion between contour passes.

Ramp angle (deg) – Upper field. This is the angle from horizontal, which the tool will be going down at in motion where the tool is not in contact with the surfaces. If this angle is large enough, then a helix (spiral) loop is created.

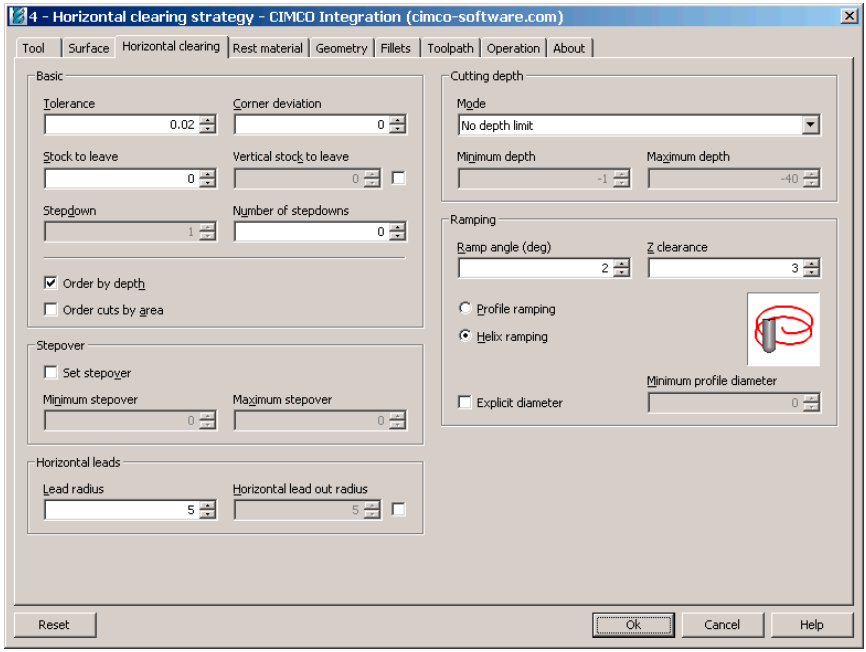
Z clearance specifies the distance above the top of stock, where the entry move begins.

Contact ramp policy is the method used to move from one contour pass to the next, if the tool is kept in contact with the surfaces. The choices are a straight line (the shortest distance between the two passes) or a ramping curve.

Ramp angle (deg.) – Lower fields. This is the angle from horizontal, which the tool will be going down at during the ramping motion if the 'Curve' ramping policy is selected.

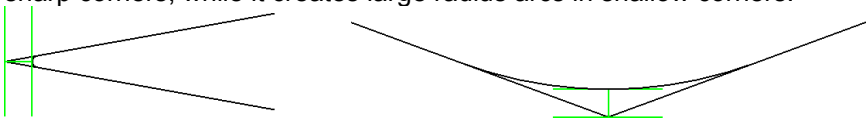
Smooth ramp: Selecting this option makes the ramping motion go in an S curve with no sharp corners.

Horizontal



Tolerance is the maximum deviation from the surface, which the contact point of the cutter will have while machining. This tolerance is before arc filtering is applied, if used. See also *Global filtering* on the operation tab.

Corner deviation describes the smoothing done in corners. To create a smooth toolpath, arcs are inserted in the corners of the toolpath, keeping the smoothened toolpath within the deviation allowance defined by the corner deviation. As can be seen below, this creates small radius arcs in sharp corners, while it creates large radius arcs in shallow corners.



Stock to leave is the amount of stock to leave on the surfaces. By checking the checkbox to the right of *Vertical stock to leave*, it is possible

to set different amounts of stock to be left in the horizontal and vertical directions.

Stepdown: The stepdown in Z per stepdown, if more than one Z level is used.

Number of stepdowns: Setting the number of stepdowns to zero will create a pass at the Z level of each horizontal area, setting it to 1 or more will create additional passes above each horizontal area.

Order by depth: Checking this checkbox will cause the toolpath to be linked in order of Z depth, so an entire Z depth is machined before the next z depth, even if there are several areas.

Order cuts by area: Checking this checkbox will cause the toolpath to be linked by closest other area. This makes for the shortest possible moves from one area to the next, but may cause areas at a deep Z level to be machined before areas at a higher Z level.

Stepover: It is possible to specify a maximum step in XY per pass, instead of allowing Mastercam HSM Performance Pack to calculate the maximum step possible while still removing all material. It is usually best to allow Mastercam HSM Performance Pack to calculate the maximum step possible while still removing all material, as it is often impossible to avoid machining with the full diameter of the tool in some areas anyway.

Horizontal leads: The *Lead radius* here is the radius of the lead in/out from the toolpath, in the horizontal plane. By selecting the checkbox adjacent to *Horizontal lead out radius*, it is possible to set a different radius for the lead out from the toolpath in the horizontal plane.

Cutting depth: Here it is possible to specify depth limits for the toolpath, useful for machining deep pockets, where the upper areas are to be machined with a shorter tool, and the lower areas with a longer tool. The depth limits are to the tip or center of the tool (as selected here), even if *Boundaries confine the tool contact points* is selected on the geometry tab.

Ramping: This describes the entry moves for each Z level.

Ramp angle (deg.): This is the angle from horizontal, which the center of the tool moves at during the entry ramping.

Z clearance: The ramping entry starts this distance above the top of the stock before the first Z level of each horizontal area.

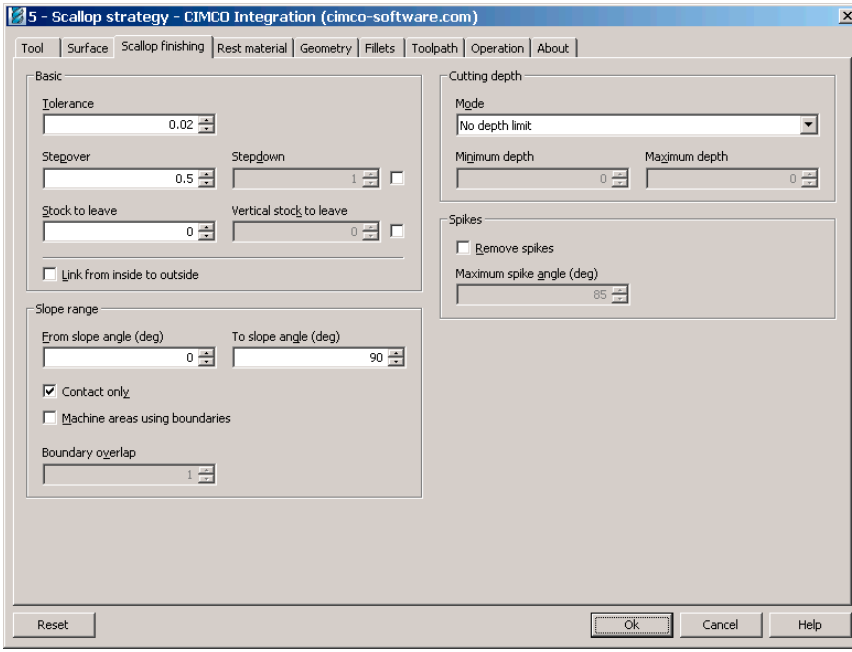
Profile ramping: The tool will follow the first pass of the next Z level as it ramps down.

Helix ramping: The tool will ramp down in a helix (spiral), where the radius of the helix is calculated automatically, based on the tool. If there is not room for a helix, then the tool will follow the first pass of the next Z level as it ramps down.

Explicit diameter: This option allows a minimum size for the entry profile to be set.

Minimum profile diameter: This is the minimum profile size for the entry ramping. It is measured along the longest direction of the profile. If *Helix ramping* is selected above, and there is not room for the helix, then *Profile ramping* is used. If *Profile ramping* is selected (or helix ramping was discarded), then the first pass of the new Z level that is larger than the *Minimum profile diameter* will be used for ramping.

Scallop



Tolerance is the maximum deviation from the surface, which the contact point of the cutter will have while machining. This tolerance is before arc filtering is applied, if used. See also *Global filtering* on the operation tab.

Stepover is the step along the surface from one pass to the next. This is measured along the surface to create a uniform surface finish on all surfaces, regardless of slope angle. By checking the checkbox to the right of *Stepdown*, it is possible to set a different (normally smaller) value for the Z direction. If the stepover would otherwise result in a stepdown in Z greater then the value set in *Stepdown*, then the stepover is reduced so the stepdown in Z is the value set in *Stepdown*.

Stock to leave is the amount of stock to leave on the surfaces. By checking the checkbox to the right of *Vertical stock to leave*, it is possible to set different amounts of stock to be left in the horizontal and vertical directions.

Link from inside to outside means the tool starts at the midpoint, machining from there out to the toolpath boundary, instead of starting at the toolpath boundary and machining in towards the midpoint.

Slope range: Only areas where the slope angle of the surfaces is within these limits will be machined.

Contact only: If this checkbox is checked, toolpath segments where the tool is not in contact with the surfaces will be filtered out. With this checkbox not checked, there will also be scallop passes along the *Maximum depth*, if depth limits are active.

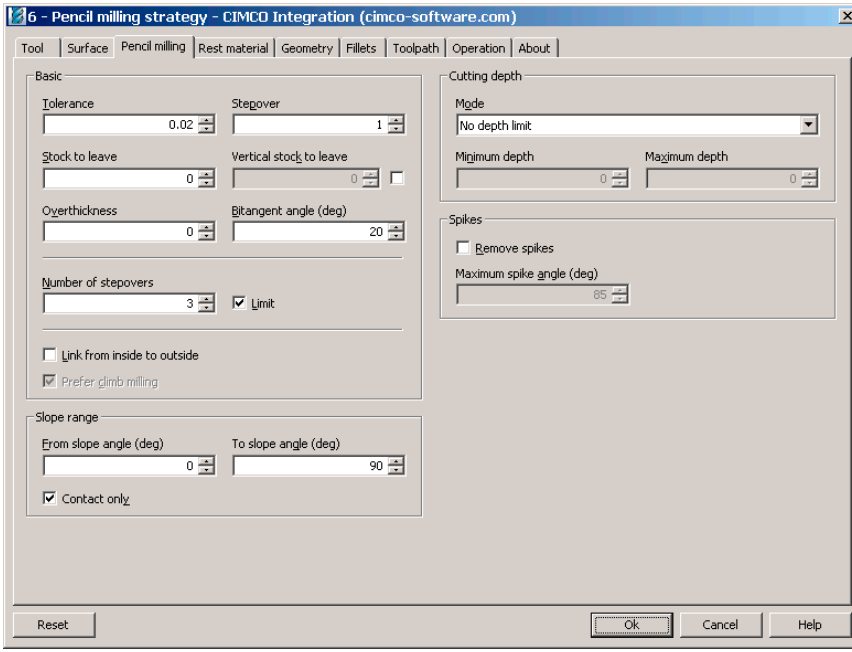
Machine areas using boundaries allow the boundaries created by the *Slope range* selected above to be offset, which in some cases creates smoother boundaries.

Boundary overlap: The distance the boundaries created by the *Slope* range selected above are offset. This offset is measured along the surfaces.

Cutting depth: Here it is possible to specify depth limits for the toolpath, useful for machining deep pockets, where the upper areas are to be machined with a shorter tool, and the lower areas with a longer tool. The depth limits are to the tip or center of the tool (as selected here), even if *Boundaries confine the tool contact points* is selected on the geometry tab.

Maximum spike angle (deg) Activating this function with the *Remove spikes* checkbox allows the user to specify a maximum angle used to remove spikes from the toolpath. The maximum angle is the direction change in the toolpath that should cause a check to remove a spike, and if the toolpath returns to the prior direction shortly after heading away on a spike, then the spike is removed.

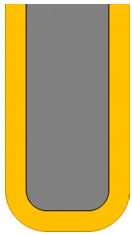
Pencil



Tolerance is the maximum deviation from the surface, which the contact point of the cutter will have while machining. This tolerance is before arc filtering is applied, if used. See also *Global filtering* on the operation tab.

Stepover is the step along the surface from one pass to the next. This is measured along the surface to create a uniform surface finish on all surfaces, regardless of slope angle.

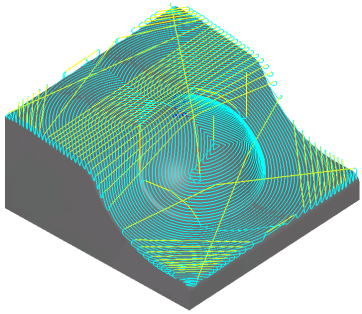
Stock to leave is the amount of stock to leave on the surfaces. By checking the checkbox to the right of *Vertical stock to leave*, it is possible to set different amounts of stock to be left in the horizontal and vertical directions.



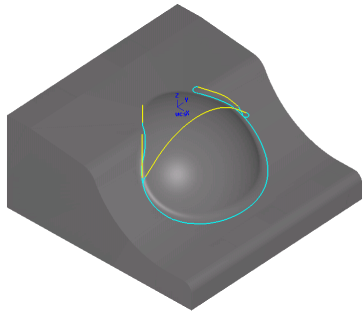
Overthickness creates the pencil toolpath as if the tool was larger by the entered amount, and then projects the toolpath on to the surfaces. This allows corners with a corner radius slightly larger than the tool radius to be machined. This overthickness is a distance from the tool profile in all directions, as shown in the image to the right. This means that a bull or end mill will also create pencil toolpaths along corners at steep sides, where the corner radius on the part is less than tool radius plus overthickness (where this tool radius is half the tool diameter, not the tool corner radius)

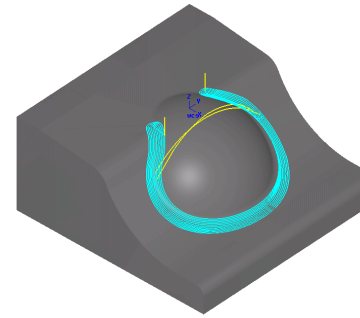
Bitangent angle is the minimum difference in angle for the tangents at the two contact points, before a pencil toolpath is generated. This is not the angle between the surfaces, but the difference between the angle between the surfaces and a straight line, so for a pencil toolpath to be created in a 150 degree corner, but not in a corner of more than 150 degrees, the bitangent angle should be set to 30 degrees. A common setting for the bitangent angle is 20 degrees.

Limit: With this option deselected, the pencil operation will create pencil parallel passes over the entire part (within the boundary), as shown to the right. With this checkbox checked, only the pencil pass itself and a limited number of pencil parallel passes will be created, as below.



Number of stepovers is the number of paths on each side of the actual pencil pass that should be added, to remove additional material left behind by earlier operations. With number of stepovers set to zero, only the pencil pass itself will be created as shown next page on the left, while a set number of stepovers creates pencil parallel passes as shown next page on the right.





Link from inside to outside means the tool starts at the midpoint, machining from there out to the toolpath boundary, instead of starting at the toolpath boundary and machining in towards the midpoint.

Prefer climb milling: This option is only available when there are no pencil parallel passes, i.e. when *Limit* is selected and set to zero. Selecting *Prefer climb milling* will create a toolpath where the machining is climb milling as much as possible, but this also often means there will be more retracts and repositionings. When *Prefer climb milling* is not selected, then toopath is created with as few retracts and repositionings as possible, but the tool will usually be performing both climb and conventional milling.

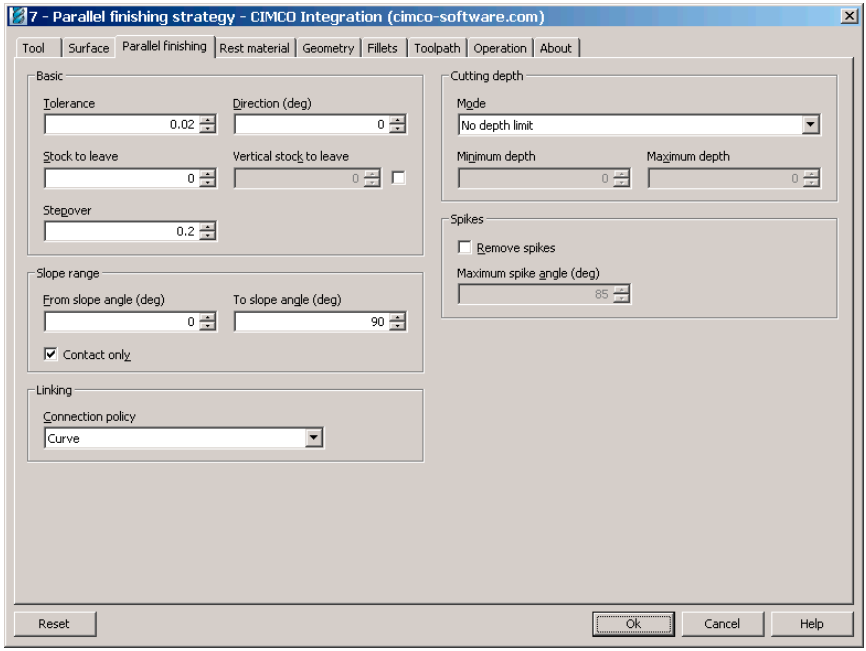
Slope range: Only areas where the slope angle of the surfaces is within these limits will be machined.

Contact only: If this checkbox is checked, toolpath segments where the tool is not in contact with the surfaces will be filtered out.

Cutting depth: Here it is possible to specify depth limits for the toolpath, useful for machining deep pockets, where the upper areas are to be machined with a shorter tool, and the lower areas with a longer tool. The depth limits are to the tip or center of the tool (as selected here), even if *Boundaries confine the tool contact points* is selected on the geometry tab.

Maximum spike angle (deg) Activating this function with the *Remove spikes* checkbox allows the user to specify a maximum angle used to remove spikes from the toolpath. The maximum angle is the direction change in the toolpath that should cause a check to remove a spike, and if the toolpath returns to the prior direction shortly after heading away on a spike, then the spike is removed.

Parallel



Tolerance is the maximum deviation from the surface, which the contact point of the cutter will have while machining. This tolerance is before arc filtering is applied, if used. See also *Global filtering* on the operation tab.

Direction is the direction of the parallel finishing toolpaths, in degrees. Parallel to the X axis from negative towards positive X is 0 degrees, parallel to the Y axis from negative towards positive Y is 90 degrees, etc.

Stock to leave is the amount of stock to leave on the surfaces. By checking the checkbox to the right of *Vertical stock to leave*, it is possible to set different amounts of stock to be left in the horizontal and vertical directions.

Stepover is the step in the XY direction from one pass to the next.

Slope range: Only areas where the slope angle of the surfaces is within these limits will be machined.

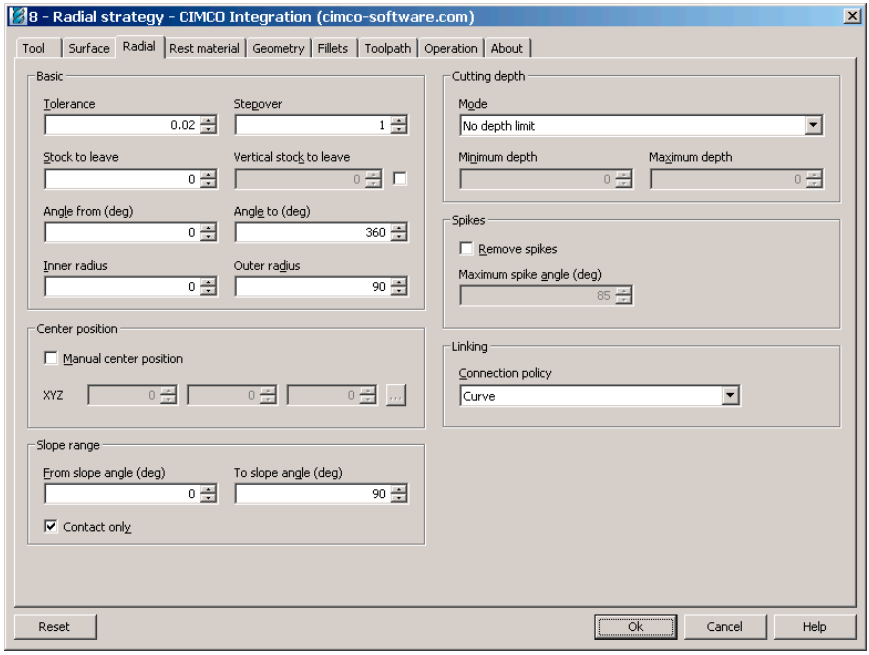
Contact only: If this checkbox is checked, toolpath segments where the tool is not in contact with the surfaces will be filtered out. With this checkbox not checked, there will also be scallop passes along the *Maximum depth*, if depth limits are active.

Linking allows the user to select whether the move between two passes should be a curve or a line. This linking is only relevant if the distance between the two passes is small enough not to be performed as a rapid retract.

Cutting depth: Here it is possible to specify depth limits for the toolpath, useful for machining deep pockets, where the upper areas are to be machined with a shorter tool, and the lower areas with a longer tool. The depth limits are to the tip or center of the tool (as selected here), even if *Boundaries confine the tool contact points* is selected on the geometry tab.

Maximum spike angle (deg) Activating this function with the *Remove spikes* checkbox allows the user to specify a maximum angle used to remove spikes from the toolpath. The maximum angle is the direction change in the toolpath that should cause a check to remove a spike, and if the toolpath returns to the prior direction shortly after heading away on a spike, then the spike is removed.

Radial



Tolerance is the maximum deviation from the surface, which the contact point of the cutter will have while machining. This tolerance is before arc filtering is applied, if used. See also *Global filtering* on the operation tab.

Stepover is the step along the outer radius from one pass to the next.

Stock to leave is the amount of stock to leave on the surfaces. By checking the checkbox to the right of *Vertical stock to leave*, it is possible to set different amounts of stock to be left in the horizontal and vertical directions.

Angle from (deg) is the starting angle of the radial toolpath. 0 degrees is the positive X direction (3 O'clock). The angle must be in the -360 degree to 360 degree range.

Angle to (deg) is the ending angle of the radial toolpath. The angle must be in the -360 degree to 360 degree range. The ending angle cannot be equal to the starting angle, to create a toolpath through a full circle, the ending angle must be 360 degrees from the starting angle.

Inner radius is the inner limit of the radial toolpath, described as a radius from the center of the radial toolpath.

Outer radius is the outer limit of the radial toolpath (this radius acts a toolpath boundary), and this is also the radius at which the stepover is measured, even if the boundaries of the toolpath otherwise keep the toolpath far inside the selected outer radius. Because of this, the outer radius is usually set only slightly larger that the actual outer radius of the toolpath.

Center position allows the user to specify the center of the radial toolpath, by selecting *Manual center position*. If *Manual center position* is not selected, Mastercam HSM Performance Pack will calculate a center position based on the boundaries of the operation.

Slope range: Only areas where the slope angle of the surfaces is within these limits will be machined.

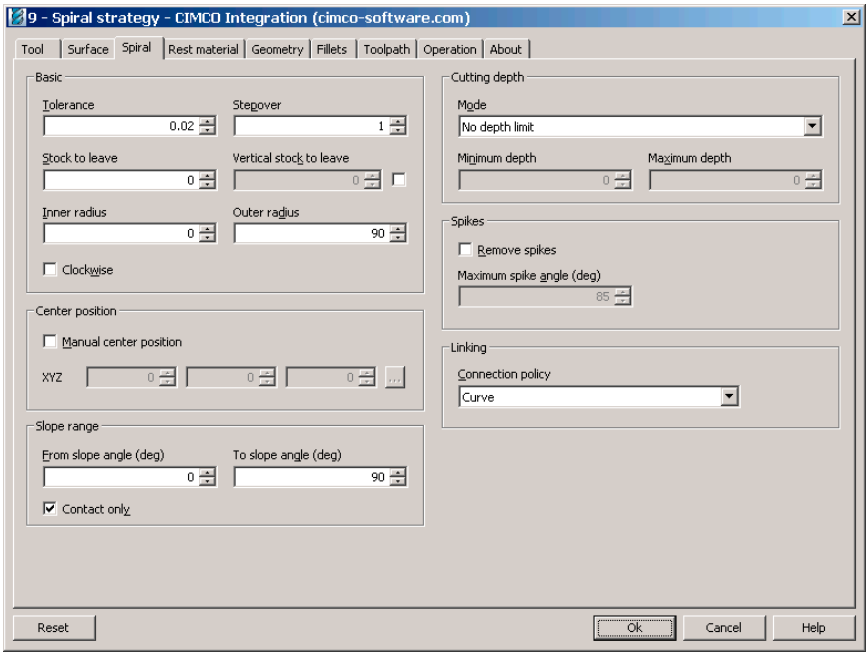
Contact only: If this checkbox is checked, toolpath segments where the tool is not in contact with the surfaces will be filtered out.

Cutting depth: Here it is possible to specify depth limits for the toolpath, useful for machining deep pockets, where the upper areas are to be machined with a shorter tool, and the lower areas with a longer tool. The depth limits are to the tip or center of the tool (as selected here), even if *Boundaries confine the tool contact points* is selected on the geometry tab.

Maximum spike angle (deg) Activating this function with the *Remove spikes* checkbox allows the user to specify a maximum angle used to remove spikes from the toolpath. The maximum angle is the direction change in the toolpath that should cause a check to remove a spike, and if the toolpath returns to the prior direction shortly after heading away on a spike, then the spike is removed.

Linking allows the user to select whether the move between two passes should be a curve or a line. This linking is only relevant if the distance between the two passes is small enough not to be performed as a rapid retract.

Spiral



Tolerance is the maximum deviation from the surface, which the contact point of the cutter will have while machining. This tolerance is before arc filtering is applied, if used. See also *Global filtering* on the operation tab.

Stepover is the step from one pass to the next, measured in the XY plane of the tool view.

Stock to leave is the amount of stock to leave on the surfaces. By checking the checkbox to the right of *Vertical stock to leave*, it is possible

to set different amounts of stock to be left in the horizontal and vertical directions.

Inner radius is the inner limit of the spiral toolpath, described as a radius from the center of the radial toolpath.

Outer radius is the outer limit of the spiral toolpath (this radius acts a toolpath boundary).

Center position allows the user to specify the center of the spiral toolpath, by selecting *Manual center position*. If *Manual center position* is not selected, Mastercam HSM Performance Pack will calculate a center position based on the boundaries of the operation.

Slope range: Only areas where the slope angle of the surfaces is within these limits will be machined.

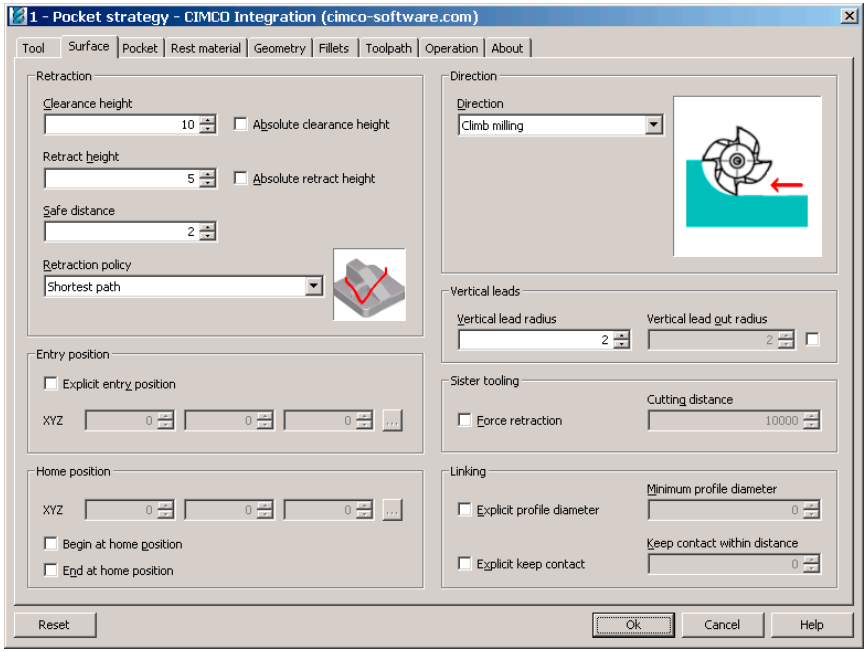
Contact only: If this checkbox is checked, toolpath segments where the tool is not in contact with the surfaces will be filtered out.

Cutting depth: Here it is possible to specify depth limits for the toolpath, useful for machining deep pockets, where the upper areas are to be machined with a shorter tool, and the lower areas with a longer tool. The depth limits are to the tip or center of the tool (as selected here), even if *Boundaries confine the tool contact points* is selected on the geometry tab.

Maximum spike angle (deg) Activating this function with the *Remove spikes* checkbox allows the user to specify a maximum angle used to remove spikes from the toolpath. The maximum angle is the direction change in the toolpath that should cause a check to remove a spike, and if the toolpath returns to the prior direction shortly after heading away on a spike, then the spike is removed.

Linking allows the user to select whether the move between two passes should be a curve or a line. This linking is only relevant if the distance between the two passes is small enough not to be performed as a rapid retract.

The surface tab



Clearance height is the height, which the tool goes to before and after the toolpath. Setting this to absolute means it is an absolute Z value, while it is otherwise an incremental height above the top of the area to be machined.

Retract height is the height where motion changes from rapid to feed. Setting this to absolute means it is an absolute height, while not setting it to absolute makes it an incremental height above the material at the place the tool is moving down.

Safe distance is the minimum distance between the tool and the part surfaces during *Shortest path* retract moves. This distance is measured after the *Stock to leave* has been applied, so if a negative *Stock to leave* is used, special care should be taken that the *Safe distance* is large enough to ensure that no collisions occur.

Retraction policy decides how moves between cutting passes are done. Shortest path is the shortest possible path (which often includes moves in all three axes), minimum retraction is straight up to the lowest height where the tool will be clear of the part (by safe Z distance), and full retraction is to clearance plane. See also *Rapid filtering* on the toolpath tab for further handling of rapid moves.

Entry position: Activating this function with the *Explicit entry position* checkbox allows the user to specify which area should be machined first, if there are several areas to be machined by the operation.

Home position: This allows the user to specify a position, which the tool must go to before and/or after the operation. This is normally only relevant when machining in 5 axis positioning, where the tool must be moved to a position from which it can move safely to the start of the next operation, but it might also be used in 3 axis machining to avoid fixtures.

Direction shows the way the machining is done, as illustrated by the image shown below the drop-down menu.

Vertical leads: The *Lead radius* here is the radius of the lead in/out from the toolpath, in the vertical plane. By selecting the checkbox adjacent to *Vertical lead out radius*, it is possible to set a different radius for the lead out from the toolpath in the vertical plane.

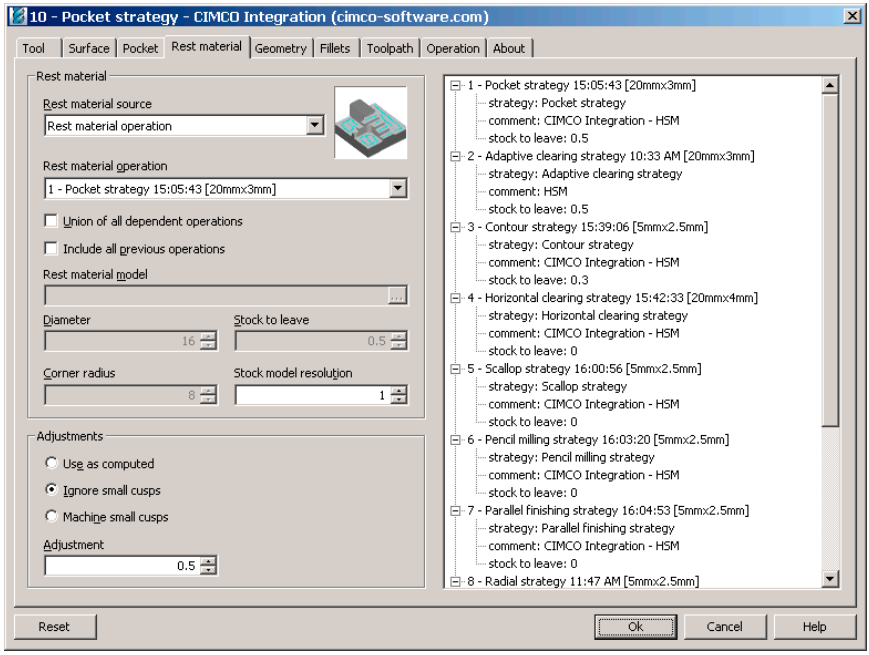
Sister tooling: Activating this function with the *Force retraction* checkbox will make Mastercam HSM Performance Pack insert a retract to *Clearance plane*, a “null tool change” NCI group and an entry to resume machining each time the tool has machined the distance selected in *Cutting distance*. Note, that the post processor must be set up to handle what should happen at this point, otherwise the tool will just retract and go back in, without changing to a different tool or allowing the operator to change inserts. This is easiest to implement using the misc. values, combined with a check of the op_id variable if desired.

Explicit profile diameter and *Minimum profile diameter*: This allows a minimum profile diameter to be set; if a machining area is smaller than the *Minimum profile diameter* then it will not be machined. The minimum profile diameter is measured as the longest distance between two points on the area boundary.

Explicit keep contact and *Keep contact within distance*: This allows the user to specify the maximum distance between two passes for contact to be maintained. If the distance between two passes is greater than the *Keep contact within distance*, then a lead out and a lead in is generated (and possibly also a retract).

The rest material tab

The rest material tab is available for the pocket, contour, scallop, pencil and parallel machining strategies. The dialog is slightly different for the pocket strategy and the finishing strategies; the dialog shown first is the dialog for the pocket strategy.



Rest material source selects the method for calculating the remaining material to be removed by this operation:

Disabled: This is not a rest material operation; the operation must be machined normally.

Rest material operation: The remaining material must be calculated as the material left behind by one or more earlier operations. This is the most correct method to use, as it works from where the tool actually went in the earlier operation, taking the room needed for lead in/out motion into account, as well as any other limitations of the earlier operations. This is also the only way to take more than one operation into account. Unfortunately, this method takes some time to calculate. The earlier operations must be selected below.

Rest material model (stereolithography): The remaining material is calculated from a STL file, defining a pre-roughed or cast part, for example. The STL file must be specified below.

Rest material tool: The remaining material is calculated from where an earlier tool could theoretically fit. This is faster than the *Rest material operation* method, but since this method does not take corner deviation and the room needed for lead in/out motion into account, or any other limitations of the earlier operation, the result is not always correct. Whether this method will give a correct result depends on the part geometry, if in doubt, it is a good idea to verify the toolpaths. The diameter and corner radius of the earlier tool, and the stock to leave used with it, must be specified below.

Rest material operation: Here the operation to be used as basis for the rest material calculation is selected, or the last of the operations if more than one operation is to be used as basis.

Union of all dependent operations: Selecting this option will use the selected operation and any operations the selected operation depends on as basis for the rest material calculation.

Union of all previous operations: Selecting this option will use the selected operation and any operations above the selected operation in the operations manager as basis for the rest material calculation.

Rest material model: Here the STL file to be used as basis for the rest material calculation is selected. Right click to browse for a file or select it from a “recent files” list.

Diameter, *Stock to leave* and *Corner radius* specifies the tool to be used as basis for the rest material calculation.

Stock model resolution is the resolution used when calculating the stock model for the rest material operation. The resolution is the size of the steps along the surfaces used to calculate the stock model.

Adjustments: It is possible to adjust the rest material model for a better result:

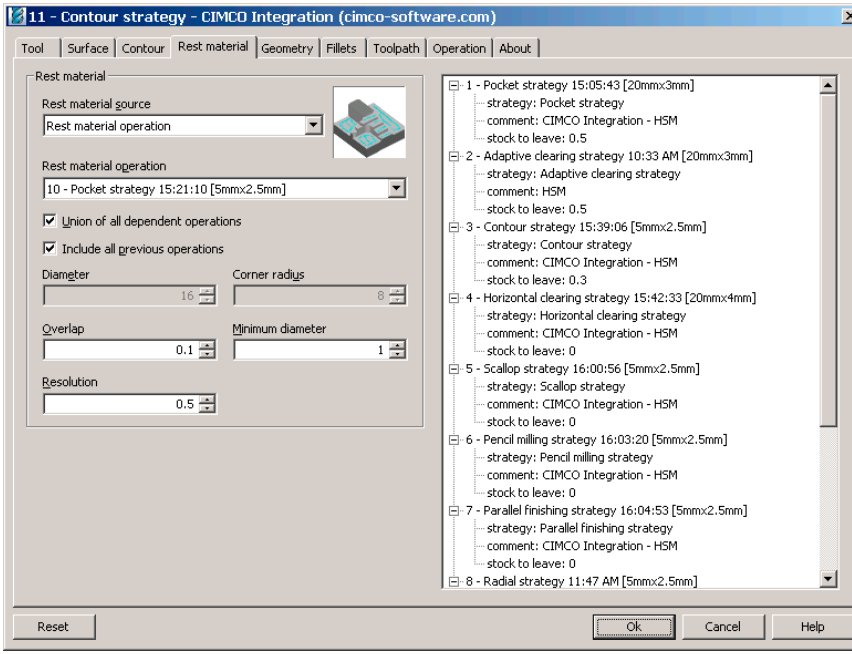
> *Use as computed* means no adjustment is used.
>
> *Ignore small cusps* causes any areas of rest material smaller than the adjustment specified below to be ignored.
>
> *Machine small cusps* extend all rest material areas by the adjustment specified below.

Adjustment is the size of the adjustment.

In the field on the right, all operations for the part created with Mastercam HSM Performance Pack can be seen. The listing for each operation shows the operation type and the tool used, and can be expanded to also show the comment and stock to leave. The listings for each operation can be expanded or collapsed by clicking the plus or minus sign next to it. By right clicking on an operation, it is also possible to view (but not change) all parameters for the operation, or expanding or collapsing all operations. When the rest material source is set to *Rest material operation*, then it is possible to select the source operation by right clicking it here and selecting *Select*. When the rest material source is set to *Rest material tool*, then it is possible to fill in the tool shape

parameters by right clicking an operation using the relevant tool, and selecting *Fill-in tool parameters*.

The rest material dialog for the finishing strategies looks like this:



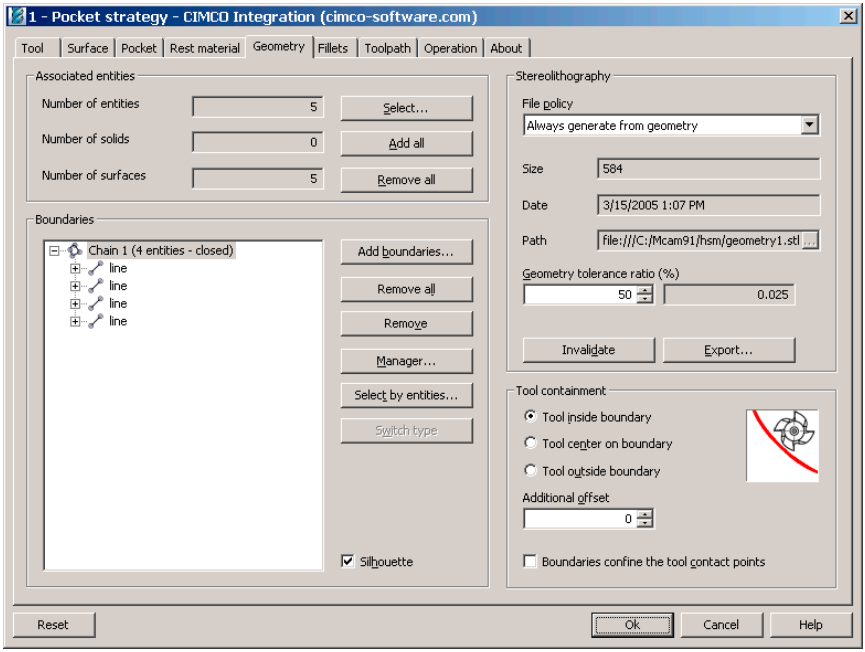
There are two fields found here that are not found on the rest material tab for the pocket strategy.

Overlap is the distance the rest material areas are extended along the surfaces to create a better surface finish. Extending the overlap will also smoothen the boundary of the rest material areas, creating smoother toolpaths, but too large an overlap will cause the boundary to become angular.

Minimum diameter defines the minimum size of a rest material area. If a rest material area is smaller than this minimum diameter, then it is not

machined. The minimum diameter is actually the size along the longest dimension of the rest material area.

The geometry tab



Associated entities shows the total number of entities associated with the toolpath, the number of these that are solids and the number of these that are surfaces.

Select allows additional entities to be selected and current entities to be unselected, while *Add all* adds all visible entities, and *Remove all* removes all selected entities.

Boundaries show the boundaries for the operation. It is possible to add boundaries with the *Add boundaries* button, and the *Remove all* button removes all boundaries in the list.

Add boundaries allows the user to add one or more boundaries to the operation.

Remove all removes all boundaries from the operation.

Remove removes the selected boundary from the operation.

Manager changes to Mastercam's chain manager, and allows the boundaries to be edited there.

Select by entities changes to the graphics window, where the user can select an entity. The boundary containing that entity (if any) is then selected.

Switch type switches a boundary between being a tool boundary and a stock boundary. This option is only available for the adaptive strategy.

Silhouette means the outer silhouette of the selected geometry is used as the outer boundary, but this will only be used if no boundaries are selected otherwise. If no boundaries are selected, and the *Silhouette* box is not checked, then an automatic outer boundary is calculated as a bounding box around the selected geometry.

Stereolithography deals with the model of triangular faces, which is used to calculate the toolpath.

File policy determines which stereolithography file is to be used, with the following options:

Always generate from geometry regenerates the stereolithography file every time the operation is regenerated. This takes a little more time, but this is the safest approach (unless using an external stereolithography file).

Reuse existing reuses the existing stereolithography for the operation, if it still exists. If the geometry has been modified, then the stereolithography file must be invalidated with the *Invalidate* button for a new stereolithography file to be generated.

Use external source means an external stereolithography file is used, usually a file generated by another program.

Invalidate causes a new stereolithography file to be recalculated at next regeneration, regardless of the file policy setting.

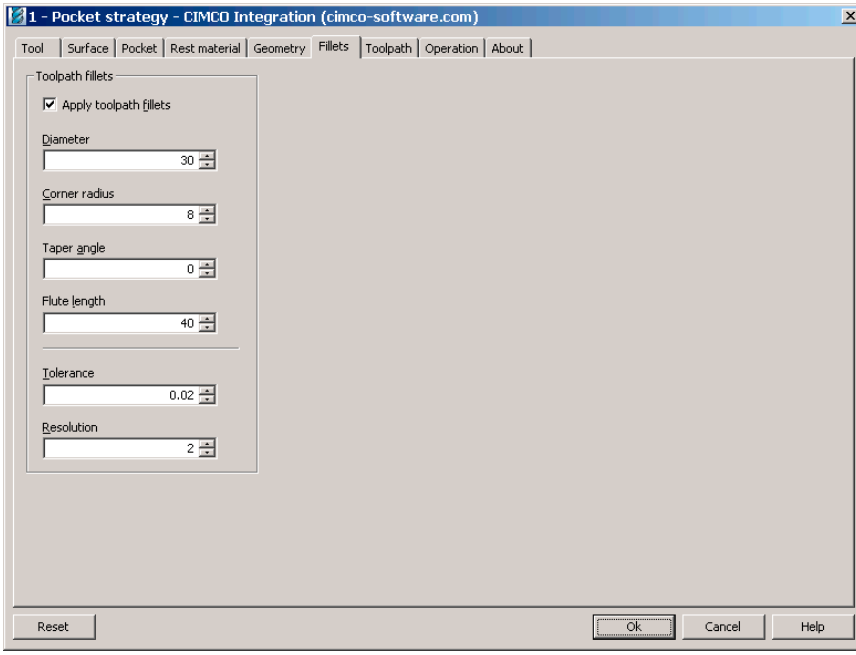
Select is used to select the external STL file, if *Use external source* was selected above.

Export exports the current stereolithography file to an external STL file for use in other programs or for comparison in verification.

Geometry tolerance ratio (%) defines the fraction of the operation tolerance (specified on the toolpath tab) used for generating the stereolithography - The remainder is used for generating the toolpaths from the stereolithography. This setting is ignored when using an external STL file. If the tool is very small, then this value should be lowered to ensure that filets are represented correctly by the stereolithography, and for complex parts, it is sometimes possible to reduce the calculation time by reducing this value.

Tool containment sets whether the tool is to be contained completely within the boundary, if it is the center of the tool that can go to the boundary, or if the tool may go to the outside of the boundary (as when machining a boss, where the boundary is the outside contour of the boss). It is possible to give an additional offset when using the inside or outside option. By checking the *Boundaries confine the tool contact points* checkbox, the boundaries are calculated according to the contact point between the tool and the surfaces, instead of the tool center. This is useful if the boundary describes the edge of the area to be machined.

The fillets tab



Apply toolpath fillets activates the toolpath filleting function. The toolpath filleting function modifies the toolpaths to prevent sharp corners in the toolpath by making the toolpath machine the area where a specified virtual tool would fit, instead of the machining surfaces. This causes fillets to be added at sharp corners and small radii.

Diameter is the diameter of the virtual tool used to calculate the filleting.

Corner radius is the corner radius of the virtual tool used to calculate the filleting.

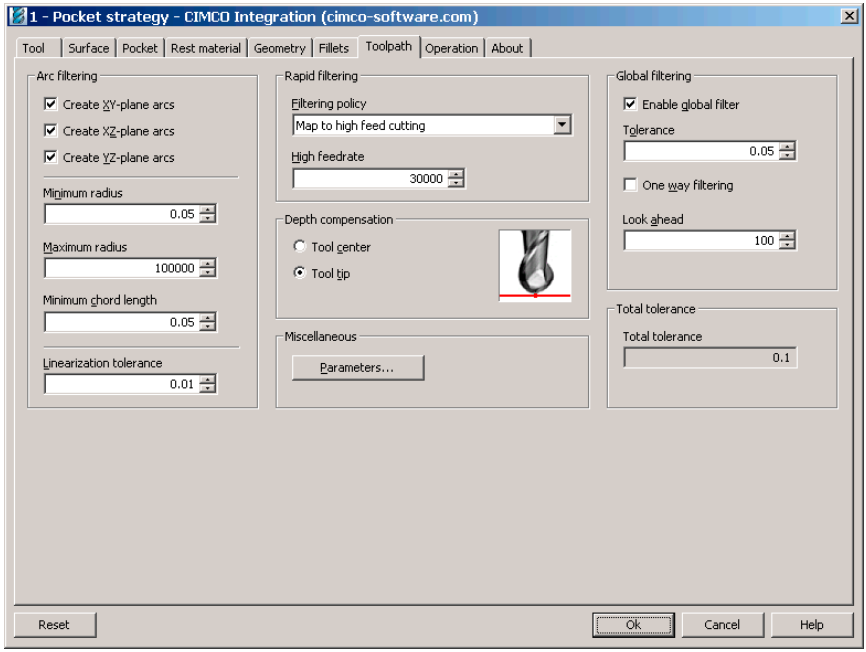
Taper angle is the taper angle of the virtual tool used to calculate the filleting. Using a taper angle on the virtual tool will make the resulting toolpath machine tapered walls in the areas where the part actually has vertical walls.

Flute length is the flute length used for the tapering described above.

Tolerance is the tolerance used for calculating the fillet areas.

Resolution is the resolution use when calculating the stock model for the fillet calculation. The resolution is the size of the steps along the surfaces used to calculate the stock model.

The toolpath tab



Arc filtering sets in which planes arcs in the toolpath can be output to the machine as actual arcs (G2/G3 in ISO code), instead of being output as line segments (G1 in ISO code). *Minimum radius*, *Maximum radius* and *Minimum chord length* are for the arcs allowed by this filtering (if the arcs do not fall within these parameters, they are converted to lines), *Liniearization tolerance* is the tolerance used when arcs are to be

converted to line segments, either because arcs are disallowed in that plane, or because the arc is too large or too small.

Rapid filtering sets when rapid moves are to be converted to linear moves (G1 in ISO code) at a high feed rate. See also *Retraction policy* on the surface tab for how retracts are generated.

There are these options:

Preserve rapid movement: Rapid moves are always output as rapid moves (G0 in ISO code), even if they are in all three axes simultaneously. This option should only be selected if rapid moves on the machine are synchronized as linear moves.

Preserve Z-axis and XY-plane rapid movement: Rapid moves are output as rapid moves (G0 in ISO code) if they are along the Z axis or in the XY plane, but are converted to linear moves (G1 in ISO code) at a high feed rate if they are in all three axes.

Preserve vertical rapid movement: Rapid moves are output as rapid moves (G0 in ISO code) if they are vertical (i.e. along the Z axis), and are otherwise converted to linear moves (G1 in ISO code) at a high feed rate.

Preserve horizontal rapid movement: Rapid moves are output as rapid moves (G0 in ISO code) if they are horizontal (i.e. in the XY plane), and are otherwise converted to linear moves (G1 in ISO code) at a high feed rate.

Preserve one axis rapid movement: Rapid moves are output as rapid moves (G0 in ISO code) if they are in one axis only, and are otherwise converted to linear moves (G1 in ISO code) at a high feed rate.

Map to high feed cutting: All rapid moves are converted to linear moves (G1 in ISO code) at a high feed rate.

High feed rate: The feed rate used when rapid moves (G0 in ISO code) are converted to linear moves (G1 in ISO code).

Miscellaneous: Clicking the *Parameters* button accesses the misc. values dialog.

Global Filtering: This deals with toolpath filtering following toolpath generation. This allows the toolpath to be further smoothened and compressed.

Enable global filtering activates the global filtering option.

Tolerance is the filtering tolerance. The filtered toolpath will be kept within this tolerance of the original toolpath. This tolerance is an additional tolerance to the tolerance on the toolpath parameter page; the total tolerance (maximum deviation from the surface after filtering) is the sum of the two tolerance values.

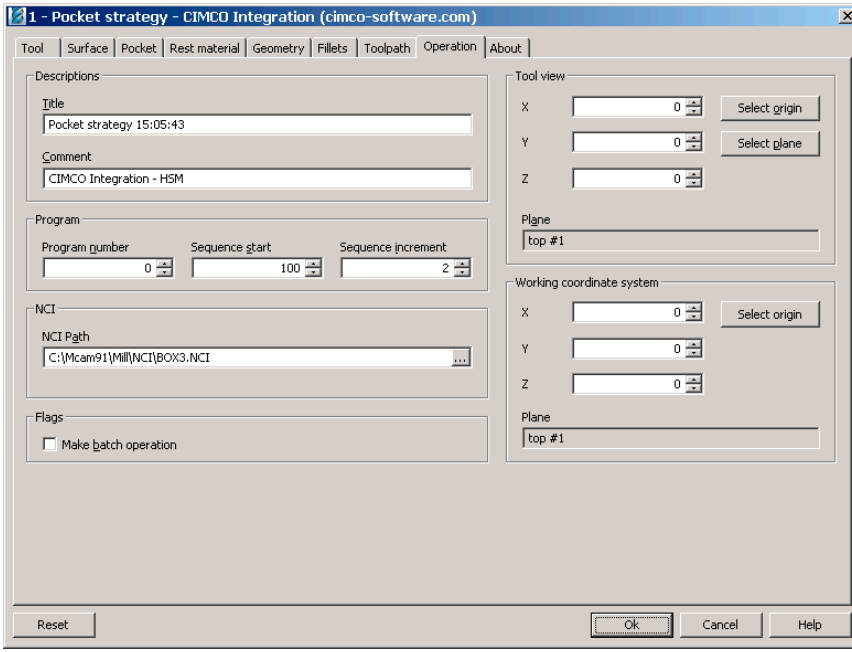
One way filtering: When a toolpath is alternating directions, such as a parallel zig-zag toolpath, then this option will make the filtering calculations run in the same direction along the surfaces. While this makes the filtering calculation take longer, it also improves the resulting surface quality.

Look ahead: This is the number of elements the filtering calculation looks ahead to see if the elements can be reduced to fewer elements. A higher number makes the filtering calculation take longer time, but may improve the reduction in file size. A value of 100 is usually sufficient to obtain optimal file size reduction without excessive calculation time.

Total tolerance lists the total tolerance for the toolpath (the sum of the tolerance from the machining strategy tab and the filtering tolerance here). The total tolerance is the maximum deviation the resulting toolpath (after filtering) can have from the original surfaces, but since the tolerance is actually the sum of three separate tolerances (Original surface to STL model, STL model to unfiltered toolpath and unfiltered toolpath to filtered toolpath), the actual deviation will usually be less than this total tolerance.

Note, that the plane settings, *Minimum radius*, *Maximum radius* and *Minimum chord length* settings under *Arc filtering* also restrict the output of the global filtering.

The operation tab



Description shows the operation description for the operation manager in Mastercam and the operation comment.

Program is the program and line numbering settings.

Make batch operation means the toolpath is not calculated now, but first when the user chooses to calculate batch operations. Using this option, the user can define a number of operations, and then let the computer calculate the toolpaths while the user goes to lunch or goes home.

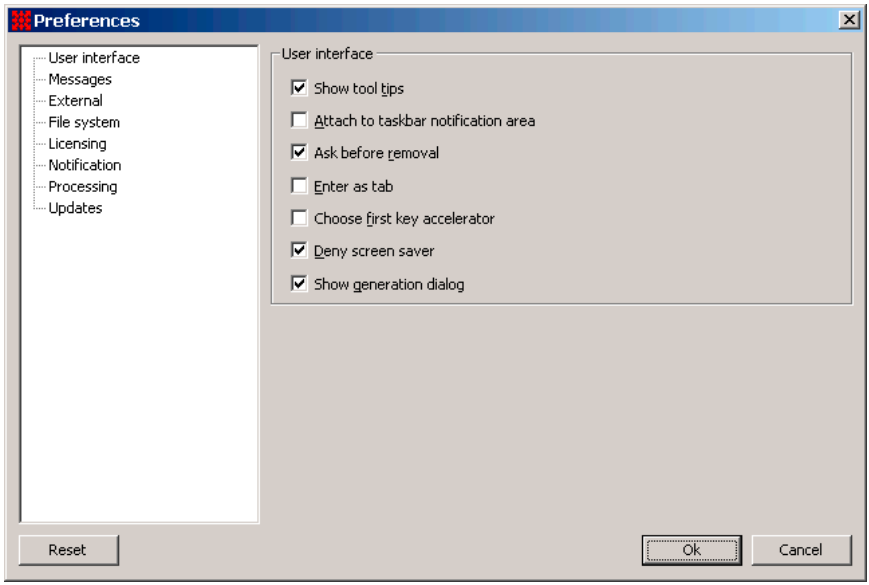
Tool view sets the tool plane and origin for 5 axis positioning.

Working coordinate system sets the WCS used in the operation.

The about tab

The *About* tab is information about Mastercam HSM Performance Pack, licensing information, and how to contact CIMCO Integration.

The options dialog



The left hand part of the options window lists the subsections of the options, the right hand part are the options for the selected subsection.

User interface

Show tool tips displays the little (usually yellow) explanations when the cursor is hovering over a field.

Attach to taskbar notification area allow Mastercam HSM Performance Pack to write notes at the task bar.

Ask before removal sets Mastercam HSM Performance Pack to ask the user for confirmation before a critical element within an operation is deleted.

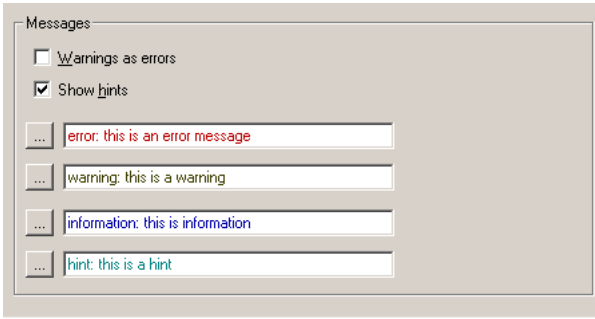
Enter as Tab makes the Enter key also function as the Tab key, jumping to the next field in the dialog.

Choose first key accelerator makes the accelerator key on the keyboard select the first occurrence of the accelerator key in the dialog when pressed. If this is deselected, the accelerator key on the keyboard will cycle between the fields in the dialog with that accelerator key.

Deny screen saver prevents the screen saver from activating while the toolpath is being calculated, so the user can see how far the progress is at all times.

Show generation dialog displays a generation dialog while the toolpath is being calculated, with all errors, warnings, information and hints, and a progress bar.

Messages



This shows the color-coding of messages in the generation dialog, and toggles these two options:

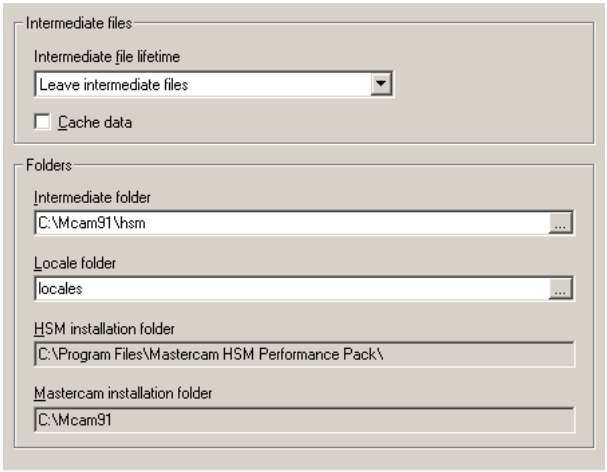
Warnings as errors: If this option is selected, warnings are color-coded as errors.

Show hints: Select this to display hints in the generation dialog and log file.

External

Editor selects the text editor to be used for displaying text files.

File system



Intermediate file lifetime determines how long intermediate files are kept.

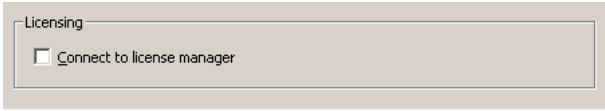
Cache data: This will cache the toolpath data, substantially shortening regeneration time if the changes to an operation do not change the path of the tool, such as if the feed rate is changed.

Intermediate folder is the folder where Mastercam HSM Performance Pack stores intermediate files.

Locale folder is the folder where the alternate language files are stored. If you are translating Mastercam HSM Performance Pack yourself or modifying the existing translations, then it is a good idea to set this to a different directory from the default directory, as your translation files might otherwise be overwritten when updating Mastercam HSM Performance Pack.

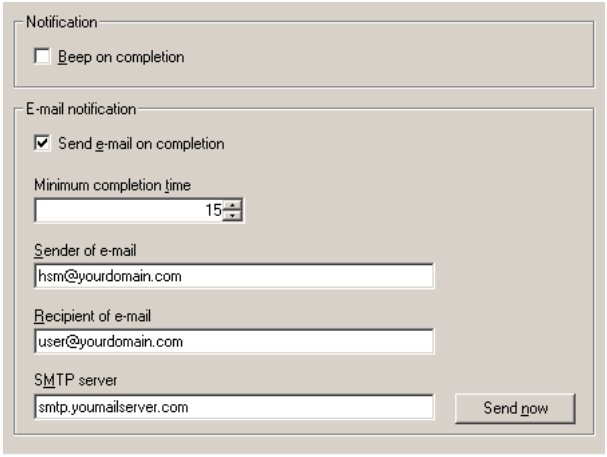
HSM Installation folder and *Mastercam installation folder* are listed for your information.

Licensing



Connect to license manager makes Mastercam HSM Performance Pack look for a CIMCO License Manager, in order to obtain a floating license for Mastercam HSM Performance Pack. This option must be checked if you are using floating licenses (only for use with a NetHASP), and must be deselected if you are not using floating licenses. For further information, see the Installation Guide for CIMCO License Manager.

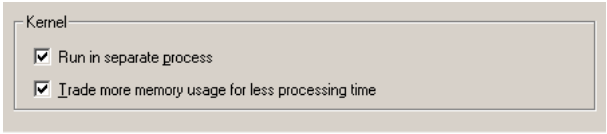
Notification



Beep on completion causes a beep when the toolpath for the operation has been calculated.

E-mail notification is used to send an e-mail to the specified e-mail address when toolpath generation has completed for an operation, if the toolpath generation took longer than the specified *Minimum completion time*. The three fields have to be filled in correctly, of course, and the *Send* button can be used to send a test message.

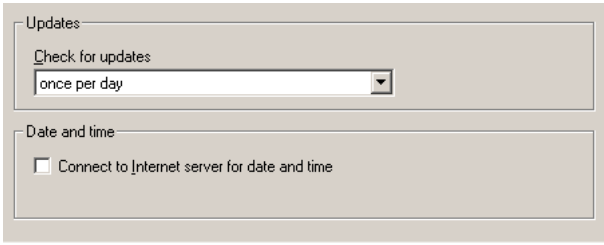
Processing



Run in separate process causes Mastercam HSM Performance Pack to run as a separate process in Windows, instead of running as part of Mastercam's process. This can speed up toolpath generation on some systems, but the user should be aware that this separate process is a legitimate process.

Trade more memory usage for less processing time speeds up processing at the cost of using more memory. It is recommended that this option is selected, unless the kernel runs out of memory.

Updates



The dropdown menu sets how checks for updates are to be made. This can be set to require manual checks, or Mastercam HSM Performance Pack can be set to perform automatic checks daily, weekly or monthly. If an automatic check detects that there is a new version available, then the user will be notified of that, and will have the possibility of downloading the new version (the same options also exists when performing a manual check). The automatic check requires that there is an Internet connection available at the time the check is scheduled; otherwise the check is not performed.

Mastercam HSM Performance Pack can be set to connect to our server to check the date and time - This can be necessary if the time on the PC has to be modified. Only licensing information is exchanged between our server and the PC running Mastercam HSM Performance Pack.

Aborting/viewing during toolpath generation

If only one operation is being generated, either because you clicked Ok in an operation dialog or because you chose to regenerate a single operation, then the toolpath generation dialog is visible.

With the toolpath generation dialog visible, it is possible to select whether the toolpath generation dialog should close automatically when the toolpath has been generated or not, and the messages from Mastercam HSM Performance Pack can be seen. It is also possible to abort the toolpath generation, and to view (but not change) the settings for the operation.

If several operations are being generated, then the progress is listed at the bottom of the Mastercam window, and there are two keys that have a special effect if the Mastercam window is the active window. Esc aborts the toolpath generation for the current operation, and Space displays the generation dialog for the current operation.